2018 US Independence Day - 3D printed memorabilia

Project Title	2018 US Independence Day - 3D printed memorabilia
Project Summary	The State Department is looking for interns to design creative, professional looking 3D-printable objects to commemorate the 2018 4th of July. The designs should be State Department and/or Independence Day themed, with a bit of technology mixed in to show our “progressiveness."
Country	United States
Country/Region of Focus	Global

Project Description

Feel free to use or incorporate emblems/graphics/seals from the State Department or US government in general following the guidelines below:
• Size: No Larger than 1” x 2” x 1”
• Support areas: if there are hollow parts and cantilevers on the model, it will increase complexity and post processing but is not frowned upon.
• Resolution: The lower the part the better. Please try to keep it below 1” in height
• Theme: State Department and/or Independence Day themed, with a bit of technology mixed in to show our “progressiveness”
• Print Settings: If the designer is familiar with 3D printing, please provide: recommended material, layer height, speed, infill thickness, support
• Single Extrusion: Uses one filament extruder
o Cannot use secondary support materials, however, it can still use the same material to print support structures
• Dual Extrusion: Uses two filament extruders for simultaneous printing of two colors/materials in the same print
o The main benefit of this is being able to use a different material for support structures so there is little to no post processing of prints.
Print Area:
• Print area is a big limitation in printing as the bigger the printers get, the more expensive and time consuming the operations get.
• In the details below, especially since the print will be of large quantity at some posts, something small like a keychain (no bigger than 1” x 2”) is probably a good size constraint
Resolution
• The main limitation with this is time. If we want something that has a nice finish, we need to increase the layers, which increases the time. For a large quantity of pieces, a piece that doesn’t need a high resolution or is fairly short would be good.
• Bonus workaround! There is a work around for low resolution high speed prints, however, it requires a small amount of post processing. This method only works with ABS, which, in itself is a special kind of hell to work

with. ABS tends to warp as it cools and requires additional print bed adhesion to heat such as painters tape or glue. (see Additional Info)

Required Skills or Interests

Skill(s)
Design thinking
Graphic design
Infographic design

Additional Information

Additional skills: Graphic Design, 3D modeling, Computer Aided Design (CAD), 3D printing.
(contd. from Project Description) o Method: The workaround is called an acetone bath. The process involves taking a small amount of acetone (nail polish remover), about enough to cover the bottom of a glass jar, covering the top of the jar with a sheet of paper or something loose, setting it on the print bed, and heating it up to 60 Celsius. This is the boiling point of acetone and it slowly becomes a vapor in the jar. Once it vaporizes, you can lower the print into the jar suspended by a string and in about 30 seconds, the acetone will melt the outside of the print and smooth the edges. Here is a video of the process:
https://www.youtube.com/watch?v=h2lm6FuaAWI
o Again for a large amount of pieces, this might suck to post process but the output is aesthetically pleasing.
Material: There are a few primary materials for printing with include PLA, ABS, Polycarbonate, Nylon (the latter two are not very common but I use them).
• Considerations:
o Ease of use: PLA wins in this regard. It extrudes nicely at high speeds and doesn't require any additional build plate adhesion techniques
o Glass Point: This is not commonly considered but it refers to the temperature at which the plastic will deform. For instance, if you are printing something for your kitchen like a measuring cup or a coffee mug and you want to wash it in the dishwasher, you would want to use something like polycarbonate as its glass point is above the boiling point of water (212F). PLA has a glass point of 170F, so if it is left on a dashboard in the sun or put in hot water, it will deform and warp and no longer hold its structural integrity. Depending on the use, this should be a consideration as materials with a high glass point are typically more expensive and tend to have a direct correlation with print difficulty.
o Sustainability: I thought I would hit on this since it is a "hot word" in the government right now. Some plastics are made from natural materials and are considered "biodegradable" in a commercial setting. The only one I know of is PLA but a quick google search could provide more insight. It is derived from corn sugar and decompose in a couple months if run through a commercial composting setting.

Print Settings:
• Infill thickness: Percentage of infill inside the part: I usually keep this low unless it is a structural part, around 20% is good.

• Wall and top/bottom thickness: The thicker this is, the less the infill needs to be in my opinion. The new version of Cura, the slicer I use for ultimaker printers has step increase which means it will increase the infill percentage as it gets closer to a flat surface to support the print as it develops. This allows a print to use less infill in large interior areas of the print. My general setting is 1mm for both walls and top/bottom thickness. It is enough so the part isn't translucent and provides good structural integrity.
• Speed: My ultimaker 3 runs pretty comfortably at 125mm/s but some other printers can't handle that speed, mainly because the extrusion and print head temperature can't keep up so you will end up with clogged print heads because the head can't head the material fast enough as it pushes through which will in turn grind a flat spot on where the material goes through the knurled feeder, which will in turn ruin the print. 70mm/s is a good standard for lower quality printers.
• Build plate adhesion: Again this is useful for some materials such as ABS and PC but requires post processing to clean up edges with a utility knife. You can add the following to a print:
o Brim: prints a single layer about 25 passes out (or user specified) to give a larger surface area on the initial layer to prevent warping on corners and such.
o Raft: builds a raft of a few layers for the print to start on. Solves the same issues as the brim does
o Skirt: runs a single line around the print. I don't really know what this does other than purges the print head for the first layers, but it looks cool so I use it.
o Considerations: I usually try to design and orient it in a way that allows the most surface area possible on the print bed so I don't need to use an additional adhesion type. This mitigates post processing.
Disclaimer:
All original documents produced for this project become the property of the U.S. Government and cannot be reproduced or retained by the designer. All appropriate project documentation will be given to the agency during and at the end of this project. The designer shall not release any information without the written consent of the project manager.

Language Requirements

None